CORED SLAB BRIDGE PIER

Cored Slab Bridge Pier is the substructure module of the existing **Cored Slab Program**. The programs are independent but interact together by using the same "*.sav" file. The **Cored Slab Program** is written in Microstation Development Language (MDL) and the **Cored Slab Bridge Pier** is written in Java. From a user's perspective, the Java interface behaves more like a Window's program than an MDL application.

The only prerequisite to run the **Cored Slab Bridge Pier** program is that a **Cored Slab** "*.sav" file must exist and contain a valid superstructure. The user must load a "*.sav" file in the **Cored Slab Main Menu** if one has not been loaded already. **Cored Slab Bridge Pier** will do nothing without a file loaded. When a "*.sav" file is loaded, all pertinent information from the superstructure is loaded along with any substructure components that were previously saved.

To begin, access the **Cored Slab Bridge Pier's** main dialog box (Figure 2) through the path shown in Figure 1.

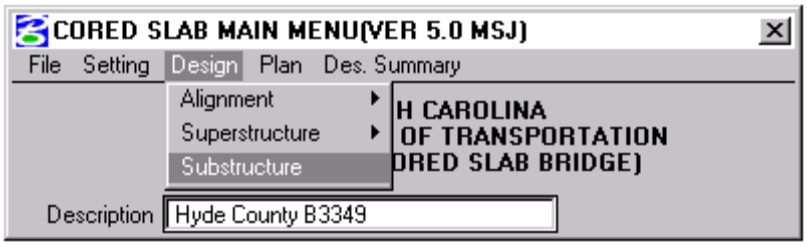

Figure 1

Cored Slab Bridge Pier Dialog Box

This is the main dialog box of **Cored Slab Bridge Pier**. The only text field is the **Title**, which is imported from the superstructure data. The text may be edited, but it will not change the "*.sav" file. The **Title** is simply transferred to each component during component creation.

The main feature in the ***Cored Slab Bridge Pier*** dialog box is the substructure tree. All substructure components and their associated data are stored here. Currently only pile end bents are available. Pile bents and post and beam bents will be added in the future.

At the bottom of the dialog box, a file path is displayed. This path represents the file that the user is working out of.

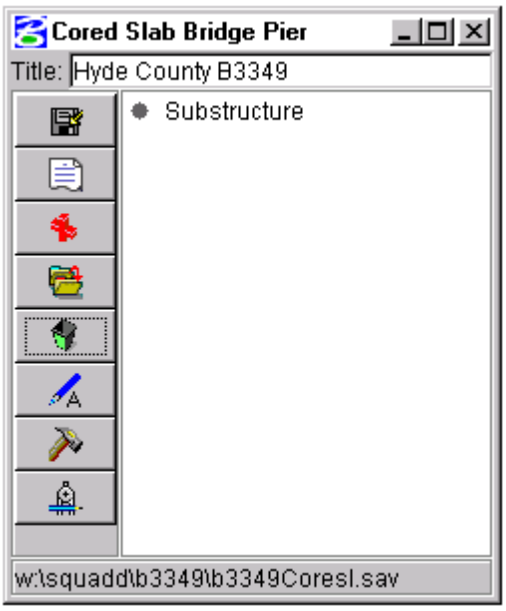

Figure 2

Along the left side of the dialog box there is a button bar with eight icons. The button bar controls are defined as follows:

 The **Save File** button saves the substructure data only to the "*.sav" file. Superstructure data must be saved in the **Cored Slab Main Menu**.

 The **List** button allows the user to open a file in the **NCBDS Editor**.

 The **Create** button opens the ***Create Pier*** dialog box.

 The **Copy** button makes a copy of an existing bridge component.

 The **Delete** button erases a bridge component and all of its associated data.

 The **Rename** button gives the user the option to rename an existing bridge component.

 The **Design** button opens the ***Design*** dialog box.

 The **Draft** button is not available at this time but will be in the future.

End Bent Design

The ***End Bent Design*** module of **Cored Slab Bridge Pier** emulates the E74900 mainframe program and performs the design or analysis of pile supported end bents. The live loads for all applicable AASHTO truck loadings are computed within the program. **Cored Slab Bridge Pier** determines the number of piles required to support all vertical loads and checks rotational stability against horizontal loads. The program also determines the reinforcing steel required to satisfy the design moments and shears in the end bent cap.

The ***End Bent Design*** module maintains all capabilities of the end bent mainframe program. The most frequently encountered input data is automatically generated by the **Cored Slab Bridge Pier** program; however, the user may exercise additional design options through the use of the **NCBDS Editor**.

For a complete description of the ***End Bent Design*** program's input and output files, see Chapter 13 of the **NCBDS User's Manual**.

Create Pier Dialog Box

The first step in designing an end bent is to create an end bent component. Select the **Create** button in the ***Cored Slab Bridge Pier*** dialog box to open the ***Create Pier*** dialog box, shown in Figure 3.

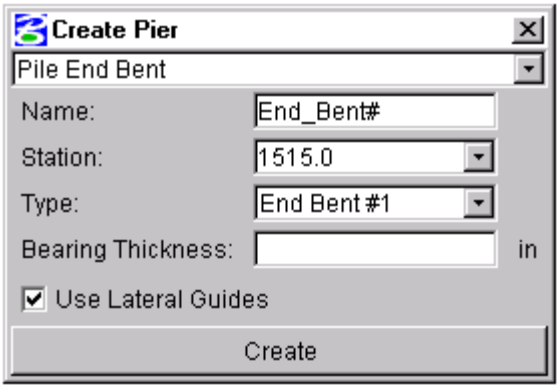

Figure 3

This dialog box is used each time a bridge component is added to a project. The ***Create Pier*** dialog box will change based on the type of component desired. Currently, the only end bent component **Cored Slab Bridge Pier** recognizes is a **Pile End Bent**. Select **Pile End Bent** from the pull down menu at the top of the dialog box.

Name:

The **Name** is the text that will be displayed in the bridge component tree of the main dialog box and the file name of the input, output, and comp files. When the ***Create Pier*** dialog box is initially opened, the **Name** field reads "End_Bent#". Place the corresponding end bent number at the end of the data field. Spaces are not allowed in the **Name**.

Station:

The pull-down menu in the **Station** data field lists the work point stations stored in the superstructure module. Select the fill face station of the end bent being designed.

Type:

Choose either *End Bent #1* or *End Bent #2*.

Bearing Thickness:

Enter the **Bearing Thickness** in this field. The thickness is used to establish the cap elevations.

Use Lateral Guides:

Activate this toggle when lateral guides are to be supplied on the end bents.

Create:

Once all of the data is entered into the ***Create Pier*** dialog box, select the **Create** button at the bottom of the dialog box to create the bridge component. Simultaneously, a file folder **Sub** and a computation file, "ComponentName.comp", for the active end bent are created. The **Sub** folder is always created in the active working directory and will house all files created by **Cored Slab Bridge Pier**. The computation file contains the generated end bend computations and takes the name shown in the **Name** field of the ***Create Pier*** dialog box.

Design Dialog Box

Once a bridge component is created, the ***Design*** dialog box, shown in Figure 4, is accessed by selecting the **Design** button in the ***Cored Slab Bridge Pier*** dialog box. The ***Design*** dialog box can only be viewed if a bridge component is selected on the substructure tree in the ***Cored Slab Bridge Pier*** dialog box.

For pile end bents, the ***Design*** dialog box consists of eight pages that combine the data into logical groups. The tabs at the top of the ***Design*** dialog box access these pages. Most of the data displayed on each page is calculated when the component is created. If any numbers are revised, the component (*.comp) file is updated when the **Recalculate** button is selected. All default values that appear when a page is accessed may be revised.

It is important to note that data displayed on the ***Design*** dialog box pages, for both end bents, is calculated and displayed as if looking up station.

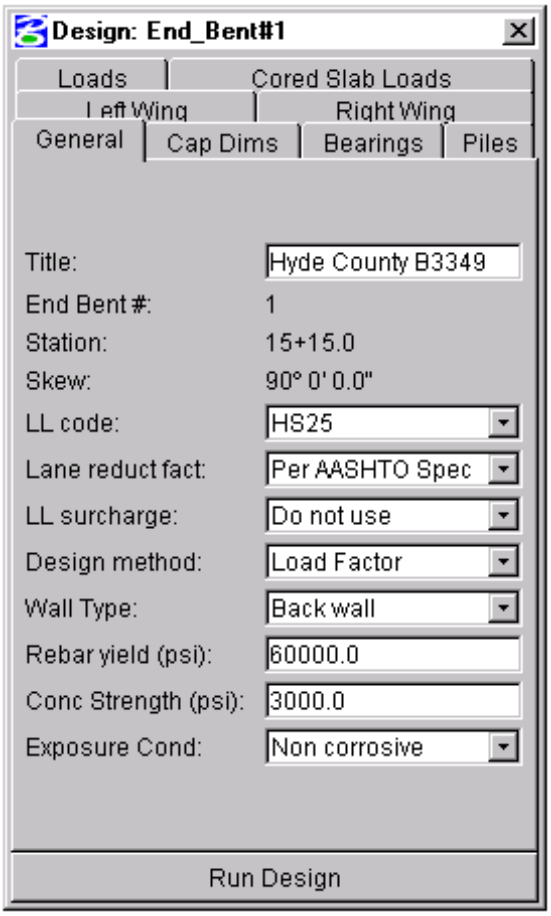

Figure 4

General

The **General** page (Fig. 4) contains the location of the end bent as well as the design options available.

Title:

The **Title** is imported from the **Description** field located on the **Cored Slab Main Menu**. The **Title** may be edited and will be saved in the "*.sav" file. The **End Bent #** and **Station** are also imported from the component data. The **Skew** is imported from the ***Geometric Definition*** dialog box in the superstructure design.

LL Code:

The **Live Load Code** is imported from the superstructure data. This is a pull down menu and the user may change the code if needed.

Lane Reduction Fact:

The user has the option to design *Per AASHTO Specifications* or to *Not Use* live load **Lane Reduction Factors**.

LL Surcharge:

The user may choose *Use* to apply a live load surcharge of 2'-0" (0.61 m) to the end bent, or *Do not use* to apply no live load surcharge.

Design Method:

Load Factor or *Service Load* may be selected.

Wall Type:

Backwall or *Curtain Wall* may be selected. **Cored Slab Bridge Pier** uses *Backwall* as the standard default.

Rebar Yield:

The *Rebar Yield* is imported from the ***Cored Slab Design Input*** dialog box in the superstructure design

Conc Strength:

Cored Slab Bridge Pier employs a default value of 3000 psi (20.7 Mpa) for the **Conc Strength** of the concrete in the end bent.

Exposure Cond:

Corrosive or *Non Corrosive* may be selected. When *Corrosive* is selected, **Cored Slab Bridge Pier** increases the concrete cap cover to the main steel and to the stirrups in the cap by 1" (25mm). A decreased Z factor is also used in the flexural distribution calculations as per AASHTO.

Cap Dimensions

The **Cap Dimensions** page (Fig. 5) contains information that defines the end bent cap. Any of the values calculated by **Cored Slab Bridge Pier** on this page may be revised. Once a value is revised, select the **Recalculate Cap Length** button located at the bottom of the dialog box. The cap length is recalculated and the component (*.comp) file is updated.

Round Length To:

This allows the user to round the overall cap length to the input increment. The **Left Length** and **Right Length** are rounded to half of this value, but **Cored Slab Bridge Pier** may round again to ensure that the total cap length is rounded to a whole increment.

Left App Offset and ***Right App Offset:***

The approach slab offset dimensions represent the distance from the control line to the right and left outside edges of the approach slab, respectively. These values are computed upon component creation and are based on the roadway width plus the rail, curb or sidewalk width.

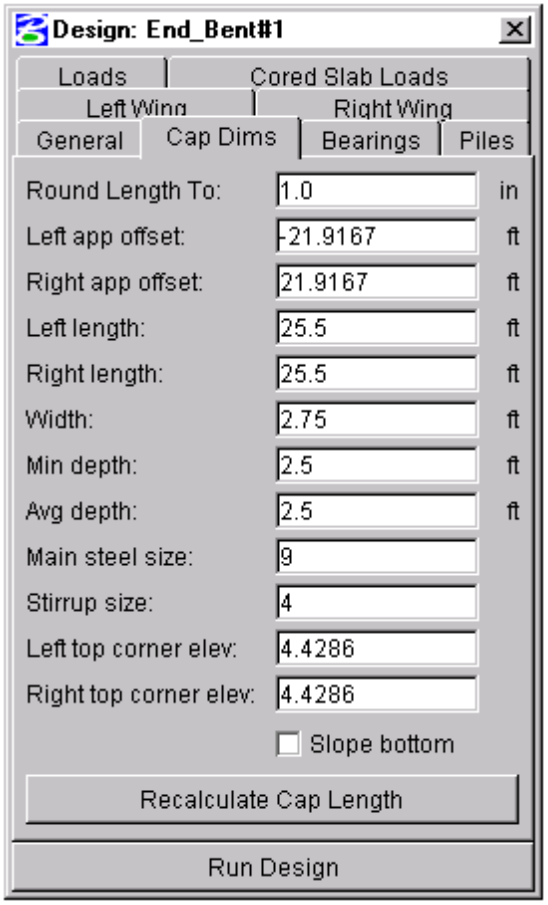

Figure 5

Left Length and ***Right Length:***

The **Left Length** and **Right Length** represent the distance along the cap from the centerline to the edge of the cap. The sum of the two equals the cap length.

Width:

The default value for the cap **Width** is 2'-9" (0.840 m). The cap width is also used to design the cap reinforcing steel.

Min Depth:

The minimum cap depth, **Min Depth**, uses a default value of 2'-6" (0.760 m).

Avg Depth:

Using the average bridge seat elevation and bottom of cap elevation, **Cored Slab Bridge Pier** calculates the average cap depth, **Avg Depth**. The **Avg Depth** is used for figuring cap weight and cap depth for application of lateral earth pressure.

Main Steel Size and ***Stirrup Size:***

For cap reinforcing steel requirements, **Cored Slab Bridge Pier** defaults to #9 (#29) bars for the **Main Steel Size** and #4 (#13) bars for the **Stirrup Size**.

Left Top Corner Elev. and ***Right Top Corner Elev:***

The **Left Top Corner Elev** and **Right Top Corner Elev** represent the elevations of the front, non-fill face corners of the end bent.

Slope Bottom:

Activate this toggle if the bottom of the end bent cap is to be sloped.

Bearings

The **Bearings** page (Fig. 6) contains information defining the elastomeric bearing pad. The user may modify these dimensions by editing the data fields in the dialog box.

Length:

The default value for the **Length** of the elastomeric bearing pad, 24" (640mm), is based on a 19" (480mm) anchor bolt gage.

Width:

The default value for the **Width** of the elastomeric bearing pad is 6" (150mm).

Thickness:

The bearing **Thickness** is imported from the ***Create Pier*** dialog box where this value was previously input.

FF to CL Brg:

The default value for the fill face to the centerline bearing is 19.5" (490mm).

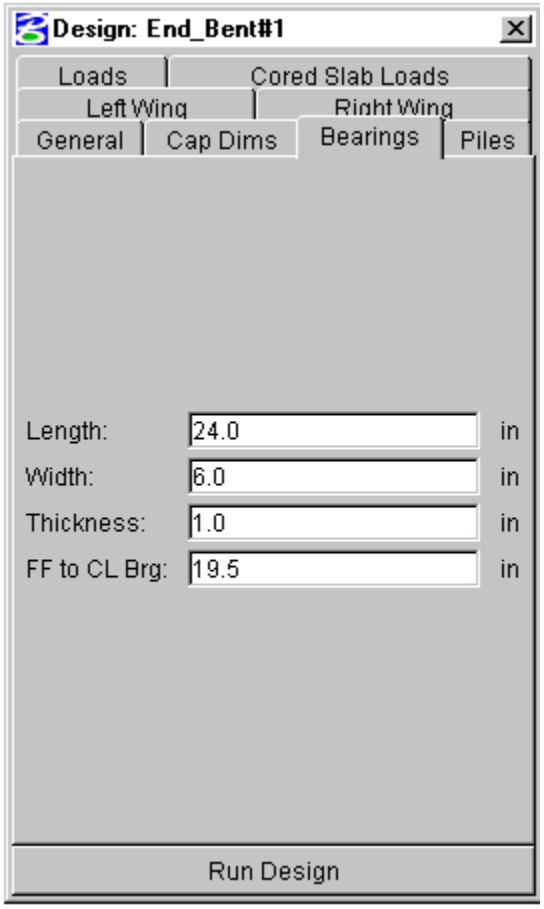

Figure 6

Piles

The **Piles** page (Fig. 7) contains information defining the end bent piles. The user may modify these values by editing the data fields in the dialog box.

Type:

Under the **Type** pull down menu, the user may choose from seven standard piles or a *Special* pile when a non-standard pile is to be used.

Capacity:

This is the **Capacity** of the pile. The default **Capacity** is 45 tons (400 kN).

Number of rows:

Either *1* or *2* rows of piles may be selected from the pull down menu.

Number vertical and ***Number braced:***

If designing, leave these values as zero and **Cored Slab Bridge Pier** will run the design option when **Run Design** is selected. If analyzing, enter the corresponding number of piles in the two data fields. When **Run Design** is selected, **Cored Slab Bridge Pier** performs an analysis. Anytime a non-zero number is entered in either of these two fields an analysis is preformed when **Run Design** is selected.

Brace pile loc:

To place brace piles inside the cap, select *Inside cap*. For special cases that require brace piles to be placed outside of the cap, select *Outside cap*. Braced piles that are placed outside of the cap do not carry any vertical load.

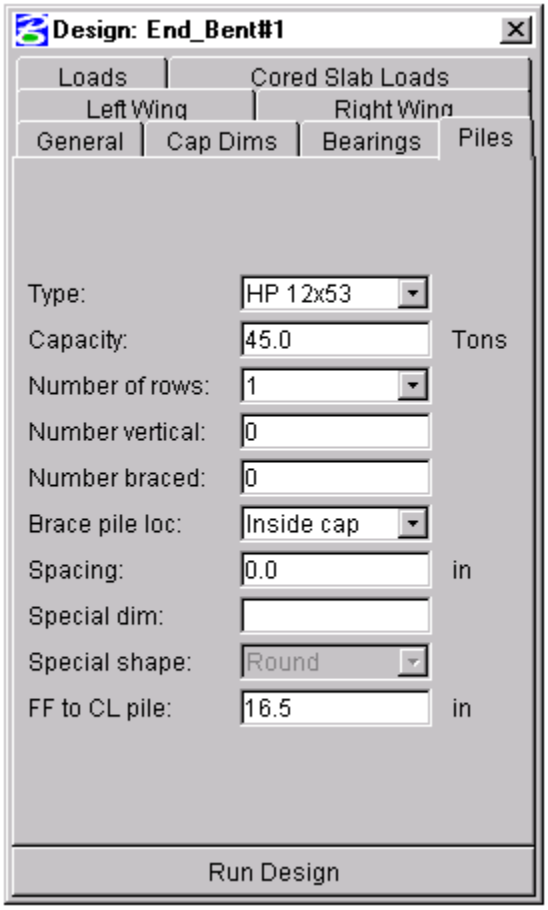

Figure 7

Spacing:

This is the **Spacing** of the piles measured along the centerline of piles. For design purposes, enter zero. For analysis, enter the pile spacing.

Special dim:

The **Special dim** is the special pile dimension. This field is only active if *special* is selected in the **Type** field. This dimension is used to calculate the clearance from the edge of pile to the end of the cap and to select positions for calculating moments and shears. For rectangular piles, enter the pile width or length, whichever is greater. For round piles, enter the diameter of the pile. If an octagonal pile is being used, enter the width between opposing faces.

Special shape:

If a S*pecial* pile is selected under the **Type** pull down menu and a **Special dim** is entered, a **Special shape**, *Round* or *Square* must be selected. If an octagonal pile is being used, select *Round* for the shape.

FF to CL pile:

The **FF to CL pile** is the perpendicular distance from the fill face to the centerline of piles. **Cored Slab Bridge Pier** uses half the cap width, imported from the **Cap Dims** page, as the default value. The user may adjust this offset if necessary.

Left and Right Wing

The **Left Wing** and **Right Wing** pages (Fig. 8) contain information that defines the end bent wings. The user may override any of the values calculated by **Cored Slab Bridge Pier** on these pages. Once a value has been changed, select the **Recalculate Wing Length** button located at the bottom of the dialog box to recalculate the cap length and update the component (*.comp) file.

Designation:

This data field alerts the user that the wing data on the page represents *W1* (wing 1) or *W2* (wing 2).

Thickness:

The default wing **Thickness** is set to 1'-0" (0.305 m).

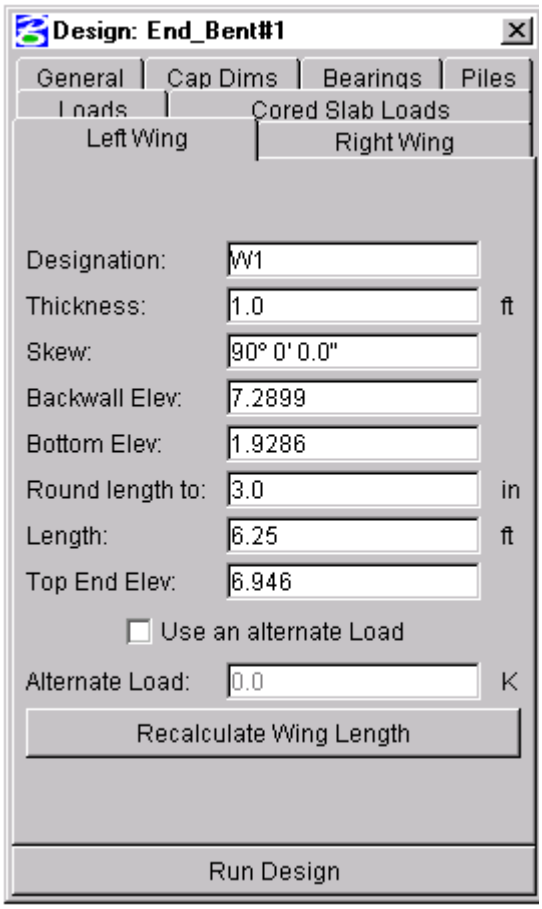

Figure 8

Skew:

The **Skew** is reported from the ***Geometric Definition*** dialog box in the superstructure design.

Backwall Elevation:

The **Backwall Elevation** is calculated based on the gutter elevation and the rail, curb or sidewalk width used. For one or two bar metal rail or a curb, this elevation is the gutter line elevation at the fill face. For a sidewalk, this elevation is the top of the sidewalk at the fill face.

Bottom Elevation:

The **Bottom Elevation** of the wing is calculated based on the bottom of cap elevation.

Round Length To:

The **Round Length To** option allows the user to round the wing length to a desired increment.

Length:

This is the calculated wing **Length** that **Cored Slab Bridge Pier** computes based on the cap geometry.

Top End Elev:

The **Top End Elev** is the top elevation at the end of the wing.

Alternate Load:

When the **Use an alternate Load** toggle is activated, the **Alternate Load** data field becomes available for input. When a load is input, an alternate wing load is applied at the end of the end bent cap. The input load will be used in lieu of the calculated value.

Loads

The **Loads** page (Fig. 9) contains loading data calculated by **Cored Slab Bridge Pier** and also requires some user input.

Wind height:

The wind area height, **Wind height,** is measured from the top of the highest barrier rail to the bottom of the lowest cored slab unit and is used to determine the wind load on the

superstructure. The **Wind height** calculation can be viewed in the component (*.comp) file.

Super depth:

The superstructure depth, **Super depth**, is the distance from the top of the end bent cap to the top of the slab at the fill face. The **Super depth** calculation can be viewed in the component (*.comp) file.

Soil fill depth:

The **Soil fill depth** is the height from the natural ground line to the crown point of the roadway at the fill face. This depth must be entered by the user and is used to determine the required number of brace piles.

Soil pressure:

The **Soil pressure** may be input or the default value of 40 lbs/ft^3 (6.3 Kn/m^3) may be used. This pressure is used to calculate the lateral earth pressure on the end bent.

Add. load and ***Add. load ht:***

The **Add. load** is the magnitude of additional load to be applied laterally at the position above the bottom of the end bent cap specified in the **Add. load ht** data field.

Sidewalk LL:

The pedestrian live load on the sidewalk, as required by AASHTO, is entered by the user as the **Sidewalk LL** and shall be input in lbs/ft per beam (kN/m per beam).

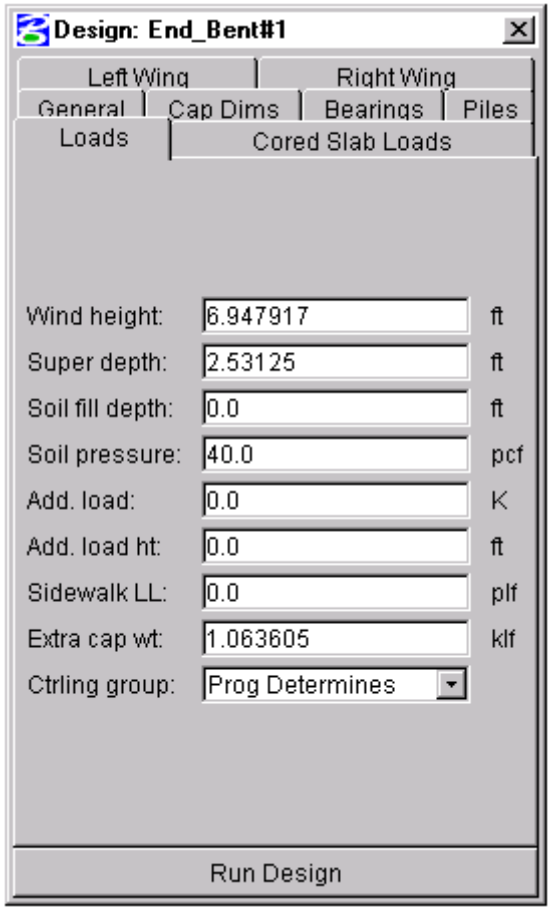

Figure 9

Extra cap wt:

The extra cap weight, **Extra cap wt** is the uniform load along the cap due to the weight of approximately half the approach slab. This calculation can be viewed in the component (*.comp) file.

Ctrling group:

The user may select group *I* through *VI* in the pull down menu for the controlling group load case or select *Prog Determines* to allow **Cored Slab Bridge Pier** to figure which group load case controls.

Cored Slab Loads

The cored slab dead loads and positions are reported on the **Cored Slab Loads** page (Fig. 10). A sample calculation for the **Load** can be found in the component (*.comp) file. The user should be aware that the ***End Bent Design*** program has a limit of 30 dead loads.

Unit #:

The **Unit #** is directly tied to the superstructure model and is not revisable. **Unit #** is the left most unit and the **Unit #** progresses from left to right.

Load:

Girder dead load reactions are calculated using loads from the cored slab design. The **Loads** in this table include the extra dead load, wearing surface, future wearing surface, the rail, and the diaphragms.

Offset:

The position of each cored slab unit, relative to the centerline of the cap (longitudinally along bridge) is calculated and presented here.

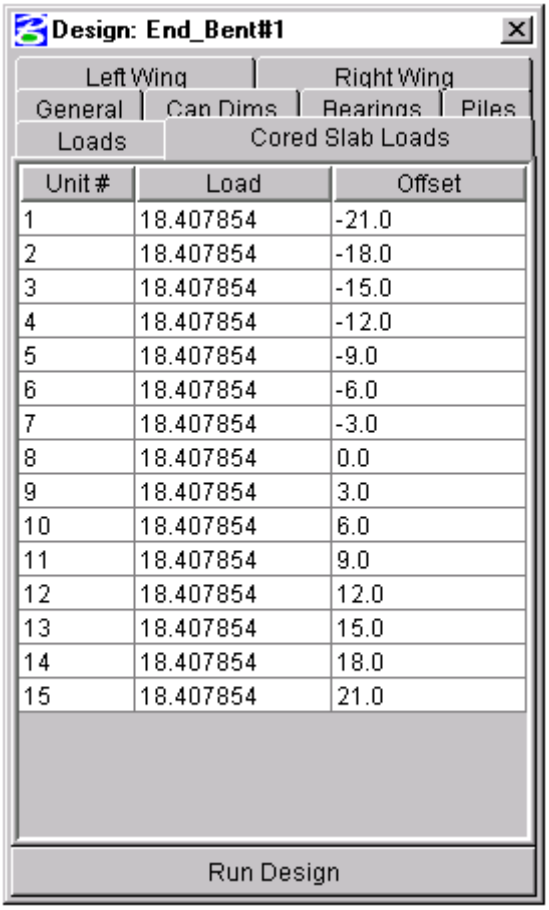

Figure 10

When all the pages in the ***Design*** dialog are complete, the ***End Bent Design*** program is ready to be executed. Select the **Run Design** button found at the bottom of any page of the **Design** dialog box to execute the program. The input file (*.inp) and output file (*.out) are created and the ***Program Progression*** window (Figure 11) is opened, when the program is executed.

At the top of the ***Program Progression*** window, the full path of the input file, the program being executed, and the output file are reported. The ***Program Progression*** dialog box reports a synopsis of the end bent design output file. The synopsis is generated by copying lines from the output file concerning the pile design and the reinforcement design. This allows the user to read the most pertinent information about a run without having to access the entire output file in the **NCBDS Editor**. The ***Program Progression*** dialog box may also be resized by simply selecting a corner of the dialog box with the cursor and dragging the window to the desired size. To exit the ***Program Progression*** dialog box, simply select the **OK** button and the dialog box will close.

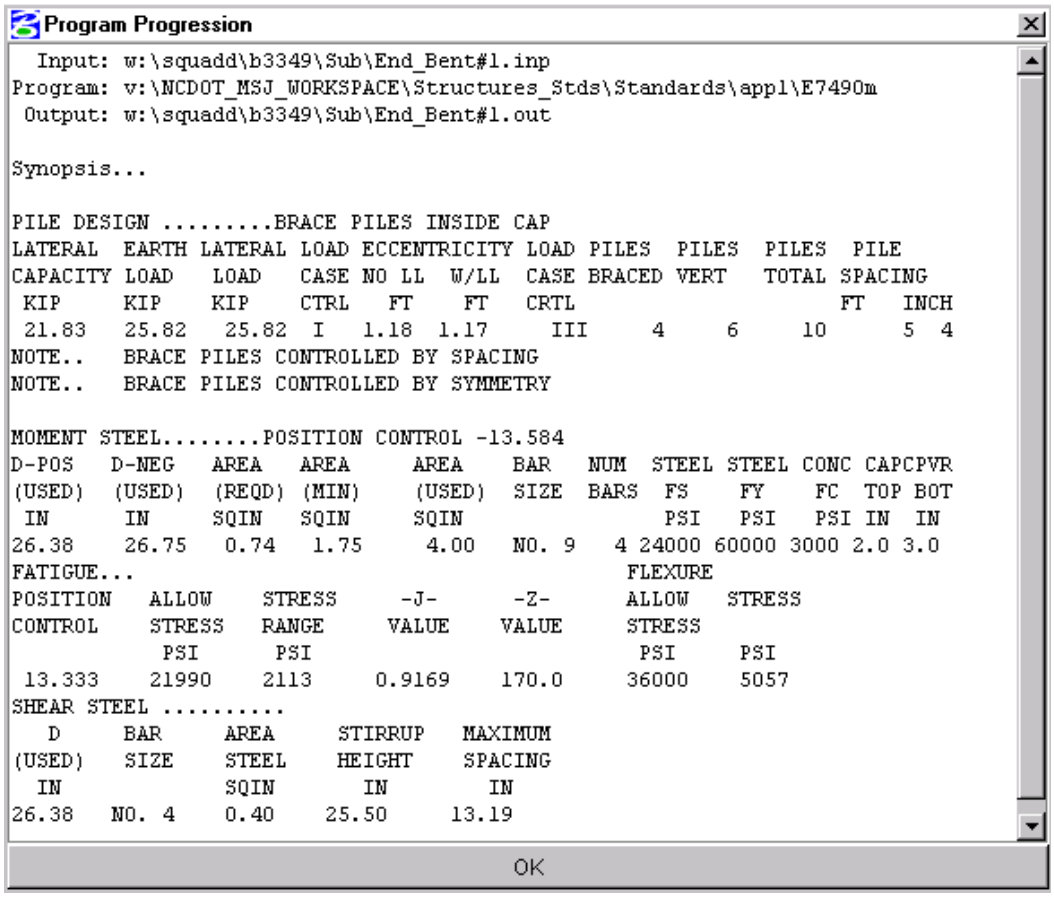

Figure 11